A SPH numerical wave basin for modeling wave-structure interactions

Wen, Hongjie; Ren, Bing; Dong, Ping; Wang, Yongxue

Published in:
Applied Ocean Research

DOI:
10.1016/j.apor.2016.06.012

Publication date:
2016

A SPH numerical wave basin for modeling wave-structure interactions

Hongjie Wen[1], Bing Ren[2], Ping Dong[3], Yongxue Wang[4]

Abstract: Based on a parallel SPH-LES model, a three dimensional numerical wave basin is developed to study wave interaction with coastal structures. The OpenMP programming technology combined with an existing MPI program contained in the parallel version of SPHYSICS code has been implemented to enable the simulation of hundred millions of particles running on a computer cluster. As part of the numerical basin development work an active absorbing wave maker and a sponge layer are introduced. The dynamic boundary conditions are also corrected to reduce the spurious effects. Wave generation and propagation in the numerical wave basin is first tested and confirmed with analytical results. Then, the model is applied to simulate wave interactions with a vertical breakwater and a vertical cylinder in order to further assess the capability of the numerical wave basin. The predicted free surface elevation near the vertical breakwater is compared with the experimental data while the horizontal forces and overturning moments acting on the vertical cylinder are verified with the analytical results. In all these cases the model results show excellent agreement with the validation data.

Keywords: SPH-LES; Numerical wave basin; Wave-structure interaction; Domain decomposition; data decomposition

1. Introduction

The hydraulic processes affecting wave interactions with coastal structures are essentially three-dimensional (e.g. wave reflection, wave transformation, overtopping and wave breaking). These processes are usually studied using a two-dimensional (2D) approach, while the flow characteristics and the consequent functional response of the structure are assessed by considering the three-dimensional effects [1]. In fact, some of the wave-induced processes such as wave diffraction around vertical cylinders and wave radiation at the breakwater head are essentially three dimensional and can only be properly described based on a 3D model. Therefore it is extremely important to develop an effective 3D numerical wave basin for modeling wave interaction with coastal structures.

In the past decades, various types of numerical wave basin have been developed using different numerical methods. Most of these methods are based on the use of mesh, such as the boundary element method [2, 3], the finite difference method [4, 5] and the finite volume method [6, 7].

Despite the great success, these grid based numerical methods suffer from various difficulties in dealing with problems with moving interface and with extremely large deformation. In addition,

[1]Ph.D, The State Key Laboratory of Coastal and Offshore Engineering, Dalian University of Technology, Dalian 116024, China. E-mail: hongjiew@126.com

2 Corresponding author, Professor, The State Key Laboratory of Coastal and Offshore Engineering, Dalian University of Technology, Dalian 116024, China. E-mail: bren@dlut.edu.cn

[3]Professor, School of Science and Engineering, University of Dundee, DD1 4HN, United Kingdom. E-mail: p.dong@dundee.ac.uk

[4]Professor, The State Key Laboratory of Coastal and Offshore Engineering, Dalian University of Technology, Dalian 116024, China. E-mail: wangyx@dlut.edu.cn

for problems with complicated geometry, the generation of a quality grid is also a difficult and time-consuming process. To avoid these shortcomings of grid-based methods, considerable effort has been made in developing meshless methods in recent decades.

Being a meshfree Lagrangian method, the SPH model does not require the explicit surface capturing scheme in treating strong nonlinear flows with large free surface deformation and enables the easy modeling of coastal structures with complex geometrical boundaries. Since Monaghan [8] pioneered the first simulation of a simple dam break problem using SPH, it has been successfully applied to fluid mechanics problems such as wave overtopping [9, 10] and wave slamming [11, 12]. Later it has been extended to solve fluid-structure interaction problems including wave interactions with caisson breakwaters [13, 14], with floating bodies [15-18] and with porous structures [19-22] as well as wave interactions with mound breakwater protected by armour blocks being discretized using SPH particles [23, 24]. Nevertheless, the studies mentioned above have all focused on two dimensional applications of SPH and few researches are confronting 3D problems due to the high computational cost of 3D SPH models except some applications concerned with dam break [25-27] and wave breaking [23, 28, 29].

Constructing a numerical wave basin usually requests a large fluid domain to avoid wave reflections from the boundaries, which means that a 3D SPH simulation would typically contains millions of particles. Therefore, a serial program is unfeasible in terms of both the memory and the simulation runtime. The parallelization of SPH code is essential for the simulation of 3D problem. Several open source codes that are specially designed for running on parallel machines have been reported, such as parallel SPHysics based on Message Passing Interface (MPI) [30] and DualSPHysics running on Graphics Processing Units (GPUs) [31, 32]. The DualSPHysics code is proven to be robust and efficient. The computing power is accelerated up to two orders of magnitude comparing with the serial version [32]. However, for large simulations that require several million particles it is essential to include an MPI implementation enabling the execution of Multi-GPUs.

The MPI is a widely accepted standard for writing message passing programs and can be employed within and across several nodes. MPI provides the user with a programming model where processes communicate with other processes by calling library routines to send and receive messages. This means the MPI program can run on a computer cluster and does not rely on a single CPU performance. However, for the MPI program the larger ratio between communication and calculation costs could be a serious drawback when the number of processors becomes large for a given problem. OpenMP is an application program interface that can be used to explicitly direct multi-threaded and shared memory parallelism within a node. Although the GPUs are more powerful than many CPU cores, OpenMP programming is much more convenient to implement with the MPI FORTRAN programming due to the ease of programming with OpenMP and the use of programming language of FORTRAN,. A hybrid MPI-OpenMP programming technology is likely to have a better parallel performance for computer clusters where MPI is used for parallelism across nodes based on spatial domain decomposition and OpenMP is used for parallelism within a node based on data decomposition.

In the present work, a viscous 3D numerical wave basin based on the SPH-LES (Large Eddy Simulation) model is established for simulating wave interaction with coastal structures. The corrective smoothed particle method (CSPM) is introduced to satisfy the normalization and symmetry conditions for boundary particles and/or irregularly distributed particles[33]. A modified dynamic boundary condition method is adopted to simulate the solid boundaries using two-layer particles in a staggered manner. In addition, The OpenMP programming technology

together with existing MPI program is developed and the computational efficiency is examined at the supercomputing center of the Chinese Academy of Sciences. The numerical wave basin is evaluated by simulating wave interaction with a vertical breakwater and with a vertical cylinder, respectively. The numerical results in terms of the free surface elevation near the vertical breakwater, the horizontal wave forces and overturning moments acting on the vertical cylinder are verified with the corresponding validation data. The characteristics of velocity and pressure field around the cylinder are also discussed.

The paper is organized as follows. After the introduction section, the 3D SPH scheme is presented, including the governing equations and the boundary conditions. In Section 3, the parallel programming is briefly described and the parallel efficiency is also tested. Section 4 describes the validation of the numerical wave basin. Sections 5 and 6 give the results of wave interaction with a vertical breakwater and a vertical cylinder, respectively. In the final section, the main conclusions are drawn.

2. Numerical model

2.1 SPH approximation techniques

In SPH, the state of a system is represented by a set of particles, which interact with each other within the range controlled by a smoothing function. Each particle carries a mass, a velocity and other properties depending on the problem, and moves according to the governing equations. Any given field function and its spatial derivative, A and ∇A, can be evaluated in the following discrete form through the kernel approximation and particle approximation [34].

$$A_i = \sum_{j=1}^{N} A_j W_{ij} V_j \tag{1}$$

$$\nabla A_i = \sum_{j=1}^{N} A_j \nabla_i W_{ij} V_j \tag{2}$$

where N is the total number of particles within the support domain of particle i. V_j is the volume associated with neighboring particle j.

In Eqs. (1)-(2), W_{ij} and $\nabla_i W_{ij}$ are the kernel function and its derivatives, respectively. However, if the original kernel function is applied directly to the approximation process, it will be unable to satisfy the normalization and symmetry conditions for boundary particles and/or irregularly distributed particles. To overcome this problem, the Corrective Smoothed Particle Method (CSPM) proposed by Chen et al.[33] based on the Taylor series expansion is implemented in the present model. In the CSPM method, the modified kernel function and its derivatives (i.e. $\tilde{W}_{ij}$ and $\tilde{\nabla}_i W_{ij}$) can be written as

$$\tilde{W}_{ij} = \frac{W_{ij}}{\sum_{j=1}^{N} W_{ij} V_j} \tag{3}$$

$$\tilde{\nabla}_i W_{ij} = \frac{\nabla_i W_{ij}}{\sum_{j=1}^{N} (\boldsymbol{r}_j - \boldsymbol{r}_i) \otimes \nabla_i W_{ij} V_j} \tag{4}$$

where $\otimes$ is the tensor product and $\boldsymbol{r}_k$ is the position corresponding to particle k.

The performance of SPH model depends largely on the characteristic of the kernel function. The

quintic kernel introduced by Wendland [35] provides a higher-order interpolation at a moderate computational cost and therefore, it is chosen as the kernel function in the present model.

$$W(r,h)=\frac{21}{16\pi h^3}(1-q/2)^4(2q+1) \qquad 0\le q\le 2 \tag{5}$$

where $q=r/h$, h is the smoothing length and r is the distance between particles i and j.

2.2 Governing equations

In this model, the fluid is assumed to be weakly compressible and the Large Eddy Simulation (LES) governing equations can be written as

$$\frac{d\rho}{dt}=-\rho\nabla\cdot\boldsymbol{u} \tag{6}$$

$$\frac{d\boldsymbol{u}}{dt}=-\frac{1}{\rho}\nabla P+\boldsymbol{g}+\upsilon_0\nabla^2\boldsymbol{u}+\frac{1}{\rho}\nabla\cdot\vec{\tau} \tag{7}$$

where $\boldsymbol{u}$ is the velocity vector, $\boldsymbol{g}$ is the gravitational acceleration. υ_0 is the laminar kinetic viscosity (10^{-6} m^2/s), $\vec{\tau}$ is sub-particle scale (SPS) turbulence stress which is equivalent of the sub-grid scale (SGS) in an Eulerian grid method. The concept of SPS is first put forward by Gotoh et al. [36] and incorporated into their Moving Particle Semi-implicit Method (MPS).

In order to close the equation, the eddy viscosity assumption is used to model the sub-particle scale turbulence stress as

$$\tau_{cd}=2\rho\nu_t S_{cd}-\frac{2}{3}\rho k\delta_{cd} \tag{8}$$

where ν_t is turbulence eddy viscosity; S_{cd} is the element of SPS strain tensor, $S_{cd}=\frac{1}{2}(\frac{\partial u_c}{\partial x_d}+\frac{\partial u_d}{\partial x_c})$, c and d refer to spatial coordinates; k is the SPS turbulence kinetic energy, δ_{cd} is the Kronecker sign function.

A modified Smagorinsky model formulated by Bradbrook et al. [37] is used to calculate the turbulence eddy viscosity and it is written as follows

$$\nu_t=\left[\min(C_s dx,\kappa l_v)\right]^2|S| \tag{9}$$

where C_s is the Smagorinsky constant and is equal to 0.1 and κ is the von Karman constant (κ=0.4). l_v is the distance from the particle to the closest boundary. dx is the particle spacing and $|S|=(2S_{cd}S_{cd})^{1/2}$. In Eq.(9), the first term in the bracket on the right-hand side of the equation governs flows far away from the solid boundary. The second term dominates for flows in the vicinity of the boundary and is used to overcome the drawback of the standard Smagorinsky model which were found to be over-dissipative near the boundary [38].

In the WCSPH method, the fluid is assumed to be weakly compressible and the pressure is calculated by the following equation of state [8]:

$$P=B\left[\left(\frac{\rho}{\rho_0}\right)^{\gamma}-1\right] \tag{10}$$

where γ is a constant and for water γ = 7 is suggested, ρ_0=1000kg / m^3 is the reference density. The parameter B can be taken as $B=c_0^2\rho_0/\gamma$ where c_0 is the speed of sound at the reference density and its value must be at least ten times greater than the maximum fluid velocity to keep density variations within less than 1% [8].

By writing Eqs. (6)-(7) in SPH form, the governing equations for particle *i* become

$$\frac{\mathrm{d}\rho_i}{\mathrm{d}t} = \rho_i \sum_{j=1}^{N} \frac{m_j}{\rho_j} \boldsymbol{u}_{ij} \cdot \tilde{\nabla}_i W_{ij} \tag{11}$$

$$\begin{aligned} \frac{\mathrm{d}\boldsymbol{u}_i}{\mathrm{d}t} = & -\sum_{j=1}^{N} m_j \left[\frac{p_i}{\rho_i^2} + \frac{p_j}{\rho_j^2} \right] \tilde{\nabla}_i W_{ij} + \boldsymbol{g} + \sum_{j=1}^{N} \frac{4\upsilon_0 m_j r_{ij} \tilde{\nabla}_i W_{ij} \cdot \boldsymbol{u}_{ij}}{(\rho_i + \rho_j)(\left| r_{ij} \right|^2 + \psi^2)} \\ & + \sum_{j=1}^{N} m_j \left(\frac{\tau_i}{\rho_i^2} + \frac{\tau_j}{\rho_j^2} \right) \cdot \tilde{\nabla}_i W_{ij} \end{aligned} \tag{12}$$

where $\boldsymbol{u}_{ij} = \boldsymbol{u}_i - \boldsymbol{u}_j$, $\psi = 0.1h$.

2.3 Boundary conditions

2.3.1 Solid boundary condition

Particles near boundaries have a support domain truncated due to the absence of neighboring particles and therefore the conditions of consistency and normalization fail. Several solid boundary conditions can be found in the literature, such as ghost boundary condition [39], repulsive boundary condition [8, 40] and dynamic boundary condition (DBC) [41]. In ghost boundary condition, fluid particles within a distance from the solid boundary are mirrored across the boundary following their local normal and the characteristics of mirrored particles (such as velocity and pressure) are determined using the fluid particle ones. This method is robust except that special attention needs to be paid to boundaries with singular geometries such as sharp angles, where a systematic mirroring of fluid particles would result in some excess of ghost mass [42]. In repulsive boundary condition, the solid boundary particles exert control through local normal forces on the fluid particles to ensure that the fluid particles can never cross the solid boundaries. However, this method cannot ensure the conservation of momentum.

In the DBC, the solid boundaries are treated by two layers of boundary particles. The boundary particles follow the same equations of continuity and state as the fluid particles, but not the momentum equations. This method is easy for specifying irregular boundaries and needs not additional programming, while a fluctuating pressure field would be produced near the solid boundaries because of the anomalously high density gradients between the fluid and boundary particles [9].

In this paper, the modified DBC method proposed is adopted. The pressure of solid boundary particle p_k is corrected by the following equation:

$$p_k = \beta p_{k,f} + (1-\beta) p_k' \tag{13}$$

$$p_{k,f} = \frac{\sum_{j=1}^{N} V_j p_j' W(r_{kj}, h)}{\sum_{j=1}^{N} V_j W(r_{kj}, h)} \tag{14}$$

$$p_j' = p_j + \rho_j \boldsymbol{g} \cdot \boldsymbol{r}_{\mathrm{kj}} \tag{15}$$

where *j* represents the fluid particle in the support of boundary particle *k*; $p_{k,f}$ is the pressure of boundary particle *k* calculated by interpolation of the fluid particles in its support domain; p_k' is the boundary pressure calculated according to the equation of state. β=0.7 is adopted in the present simulations based on extensive calibration tests. A detail discussion of the value of β can be found in Ren et al. [20].

2.3.2 Wave maker boundary condition

The wave maker boundary is simulated by a numerical plate-type wave maker that induces the motion of water to generate the required waves as well as actively absorb the energy of the reflected waves approaching the wave make that may be generated from the interior of the numerical wave basin. As shown in Fig.1, the wave maker is composed of multiple push-plates. Each push-plate is 5 times of initial particle space both in *x*-direction and in *y*-direction. The treatment of wave maker particles is the same as that of fixed boundary particles except that their positions get updated according to an external wave maker function. The numerical absorbing wave maker is developed based on the linear wave-maker theory [43]. The horizontal velocity of the wave maker can be expressed as

$$U_i(t) = \frac{\omega}{W}\left[2\eta_0 - \eta_i\right] \tag{16}$$

where ω are the frequency of the wave maker, η_0 is the target wave surface and η_i is the instantaneous wave surface at the push-plate *i*. *W* is the transfer function and can be written as $W = \frac{4\sinh^2 kd}{2kd + \sinh 2kd}$, where *k is* the wave number and *d* is the water depth.

2.3.3 Downstream boundary condition

The downstream boundary condition is designed to remove the unwanted waves during wave simulating and implemented through introducing an artificial viscosity sponge layer or damping zone at the far end of the basin. An additional artificial viscosity term is added to the momentum equation for the damping zone and the momentum equation becomes

$$\frac{d\mathbf{u}}{dt} = -\frac{1}{\rho}\nabla P + \mathbf{g} + \upsilon\nabla^2\mathbf{u} + \frac{1}{\rho}\nabla\cdot\vec{\tau} + \psi_{ij} \tag{17}$$

where ψ_{ij} is the added viscosity term and can be discretized to the following form

$$\psi_{ij} = \begin{cases} \alpha\dfrac{x_i - x_0}{L_s}\displaystyle\sum_{j=1}^{N} m_j h\frac{\bar{c}_{ij}}{\bar{\rho}_{ij}}\frac{\left|\mathbf{u}_{ij}\cdot\mathbf{r}_{ij}\right|}{r_{ij}^2+\eta^2}\tilde{\nabla}_i W_{ij} & \mathbf{u}_{ij}\cdot\mathbf{r}_{ij} \le 0 \\ 0 & \mathbf{u}_{ij}\cdot\mathbf{r}_{ij} > 0 \end{cases} \tag{18}$$

where x_i, x_0, and L_s are the abscissa of particle *i*, the abscissa of the initial position of the sponge layer, and the length of sponge layer, respectively. Moreover, $\bar{c}_{ij} = (c_i + c_j)/2$, $\bar{\rho}_{ij} = (\rho_i + \rho_j)/2$. α is a free parameter that needs to be carefully adjusted to achieve the desired wave damping effects. If the value of α is too small, the wave energy would not be absorbed completely and a reflecting wave would form from the far end of the sponger layer. If the value of α is too large, the damping zone would act in a similar way as the wall boundary and the wave would reflect from the upstream edge of the sponge layer. Following Ren et al.[20], α =0.6 is used as it gives the best wave damping results.

2.3.4 Periodic open boundary condition

There are two methods for simulating the lateral boundaries of the numerical wave basin. One is taking the lateral boundary as the solid boundary and using the DBC to calculate the pressure of solid boundary particles, and the other is using the periodic open boundary. In this work, periodic open boundary is implemented at the lateral boundaries of the wave basin. As can be seen in Fig.2, using the period open boundary, the support domain of fluid particle *a* lying near the lateral

boundary will extend beyond the lateral boundary. The missing support domain for particle *a* is continued through to the other lateral boundary. This means the fluid particles near a lateral boundary interact with the fluid particles near the complementary lateral boundary on the other side of the domain[44].

3. Parallel algorithm

The WCSPH model is solved explicitly so it lends itself for parallel programming. However, due to the Lagrangian nature of SPH method, the positions of SPH particles can change constantly. During the computation some particles may leave the resident processor and enter the neighbor processor. Compared with the parallelization strategy of mesh-based methods for which a fixed grid is employed, the parallelization of SPH model has the difficulty in accomplishing the necessary transfer of information between adjacent processors.

The parallelization strategy used in this paper combines spatial domain decomposition and data decomposition[29]. The spatial domain decomposition is performed on different processors through MPI programming. The data decomposition is implemented on each processor with shared memory architectures using OpenMP directives. Fig.3 gives the hierarchical structure of hybrid MPI-OpenMP programming.

Based on the characteristics of the numerical wave basin, the fluid domain is divided into parallel sub-domains along the direction of wave propagation (see Fig.3). In this way there is just one direction of communication between adjacent processors and the parallelization algorithm is greatly simplified. For more details about MPI parallel design the readers can refer to User guide for parallelSPHYSICS v2.0 [30].

Fig.4 displays the parallel efficiency (E_f) obtained by increasing the number of MPI processors for different problem sizes. The problem size is represented by the number of particles (n). The parallel efficiency is defined as $\frac{T_1}{N * T_N}$, where T_1 is the time taken for serial code and T_N is the time taken using N cores. The cases tested in this section are for the problem of wave propagation in the rectangular basins with different length. The simulations shown in this section have been performed at the supercomputing center of the Chinese Academy of Sciences. Each blade node contains two ten-core Intel Xeon E5-2680 V2 processors with a clock speed of 2.8 GHz and 64.0 GB RAM.

It can be seen from Fig.4 a smaller problem size corresponds to a lower parallel efficiency for the same number of MPI processors and a large number of MPI processors also correspond to lower parallel efficiency for the same problem size. It can be concluded that the parallel efficiency declines with the smaller sub-domain because under this condition the ratio between communication cost and calculation cost is larger. Therefore for a given case, it will lead to the waste of computing resource by purely increasing the number of MPI processors. It is worth mentioning that the parallel efficiency is greater than 100% for the case of n=4.0×10^6 when the number of cores is 20 or 40, which indicates the super-linear speedup. One possible reason for the super-linear speedup is the cache effect resulting from the different memory hierarchies of a modern computer: in parallel computing, not only do the numbers of processors change, but so does the size of accumulated caches from different processors. With the larger accumulated cache size, more or even all of the working set can fit into the caches and the memory access time can reduce dramatically, which results in the extra speedup in addition to that from the actual

computation[45].

OpenMP is a parallel programming language for multi-thread with shared memory architectures and has an advantage in treating interaction loops that represents the largest part of the computational costs in SPH scheme. In this paper, the data decomposition of particle interaction loops through OpenMP directives is implemented on each MPI processors.

The comparisons of parallel efficiency for the varying combination between MPI processors and OpenMP threads using 80 cores are shown in Fig.5. N_{MPI} and N_{OpenMP} represent the number of MPI processors and OpenMP threads used in the test, respectively. $N_{\mathrm{OpenMP}}=1$ means the pure MPI programming is enabled. From Fig.5 it can be seen that the hybrid OpenMP-MPI implementation can get better parallel efficiency than the pure MPI programming model and there is a positive correlation between the parallel efficiency and the number of OpenMP threads within a processor.

4. Validation of the numerical wave basin

This section is to verify the developed numerical wave basin. The length of the numerical wave basin is 12.0 m and the width is 2.0 m. The initial still water depth is 0.5 m. The wave maker is located at x=0.2 m and an artificial viscosity sponge layer of 3.0 m long is placed on the right end of the basin as shown in Fig.1. The incident wave height and wave period are H=0.12 m and T=1.2 s. A uniform particle spacing dx=2 cm and a constant smoothing length h=1.5dx are used. The fluid domains are modeled using nearly 1.5 million particles. The simulated time is 17 s and the computation time on 64 CPUs is about 2 days.

Fig.6 exhibits a 3D snapshot of the numerical wave basin at t = 8 s. Fig.7 shows the time-history profiles of wave surface collected at a series of measuring points. The coordinates of the measuring points are displayed in Table.1. Fig.7(a)-(c) can be seen that the waves rapidly reach a steady state and the wave surface profiles maintain a uniform shape in both the x and y directions. Moreover, the wave surface profiles exhibit the typical non-linear features, such as higher and narrower crests as well as smaller and flatter troughs. The analytical wave surface profiles of 2nd order Stokes waves are also displayed for comparison. As can be seen in Fig.7, the computed results have a good agreement with the analytical results. The computed wave heights at x=11.5 m are very small with most of the wave energy being damped out by the artificial viscosity sponge layer.

Besides the wave heights, the wave pressures and wave velocity are also calculated to verify the numerical wave basin. Fig.8 and Fig.9 show the computed wave pressure and wave velocity profiles compared with the analytical results at different measuring points. Table.1 displays the coordinates for the measuring points of pressures and it is noted the measuring points displayed in Fig.8 (c) are located at the free surface. The comparison clearly shows that the computed wave pressure and wave velocity profiles agree well with the analytical results.

5. Wave interaction with vertical breakwater

In the present section, the 3D SPH model is verified by considering a vertical breakwater test in a wave basin. The numerical setup follows Lara et al.'s [46] experiment as showed in Fig.10. The wave basin is 14.0 m long and 0.585 m wide with the water depth of 0.25 m. The wave maker is located at x=0.3 m and an artificial viscosity sponge layer of 3.0 m long is placed towards the end of the basin. An impervious vertical breakwater was placed in the basin. The breakwater is 0.24 m

in the *x*-direction and 0.3 m in the y-direction. The seaward face of the breakwater is located 8.3 m away from the left boundary of the basin. Following the experimental setup, the modified dynamic boundary rather than the periodic open boundary condition is implemented at the front and back boundaries of the basin in this case.

The regular wave used in the test has a wave height 6 cm and wave period 2 s. An initial particle spacing of dx =1 cm is used and the fluid domains are modeled using nearly 2.1 million particles. The simulated time is 15 s and the computation time on 64 CPUs is nearly 4 days.

Totally ten measuring points are set near the breakwater in order to capture the three dimensional effects of the waves around the breakwater better. The locations of the measuring points are shown in Fig.11. The comparison of free surface elevation between the experimental data and SPH results at different measuring points are given in Fig.12. As can be observed in Fig.12, except a slight overestimation of wave crest in WG.3, the SPH results compare with the experimental data well. Wave reflection and diffraction caused by the breakwater, can be seen from the free surface profiles of WG.4 and WG.7, respectively. A local increase of wave height is observed clearly in WG.4, which is caused by the reflection of the incident wave on the seaward side of the breakwater. The wave diffraction around the breakwater head is also well captured, which can be seen from the surface profile of WG.7.

6. Wave interaction with a vertical cylinder

6.1 Computational setup

As a further demonstration of the capability of the wave numerical basin, the interactions between waves and a vertical cylinder are also presented here. The numerical setup is shown as Fig.13. The numerical wave basin is 12.0 m long and 5.0 m wide. The wave maker is located at x=0.3 m and an artificial viscosity sponge layer of 3.0 m long is placed on the right end of the basin. The radius of the vertical cylinder is a=0.23 m. The center of the cylinder is 6.6 m away from the left boundary and in the middle of the y-direction.

Two different incident wave conditions are tested with the wave height being H=6 cm for case A and H=10 cm for case B. The wave period is 1.2 s and the water depth is 0.5 m for both cases. The parameter $2a/L = 0.225 > 0.2$ means the wave diffraction is important, where L is the wave length. An initial particle spacing of dx =2 cm is used for both cases and the fluid domains are modeled using nearly 4 million particles. The simulated time is 20 s and the computation time on 64 CPUs is nearly 5 days.

6.2 Model verification with case A

For case A, the parameter $\frac{Hk}{2th(kd)}$ is equal to 0.1, where k is the wave number. The influence of nonlinearity effect is weak and therefore the linear diffraction theory of MacCamy and Fuchs [47] can be used to verify the SPH results for case A.

The comparisons of the total forces and overturning moments acting on the vertical cylinder are shown in Fig.14. The total wave forces and overturning moments are normalized by $\rho gHa^2/2$ and $\rho gHda^2/2$, respectively. It is noted that the total wave force on the structure is calculated by means of integrating the pressure around the cylinder. The shear force, which is normally two orders of magnitude smaller than the pressure force, is neglected. It can be seen that the SPH results agree well with the analytical solutions except some small oscillations at wave troughs,

which are likely caused by the slight pressure oscillations of fluid particles around the vertical cylinder.

6.3 Model application with case B

For case B, the parameter $\frac{Hk}{2th(kd)}$ is equal to 0.17. Fig.15 gives the comparisons of the total forces and overturning moments acting on the cylinder between the computed results and analytical solution. As can be seen in Fig.15, the maximum values of the computed wave forces and overturning moments exceed the linear analytical results of MacCamy and Fuchs [47] by 12% and 15% on average, respectively. The influence of nonlinear effect can not be ignored and the linear diffraction theory cannot be used in this case.

Fig.16 depicts the time history of wave run-up on the seaward and leeward sides of the cylinder. The time history of the incident wave at the corresponding measured points (when there is no cylinder) is also displayed. From Fig.16 it can be observed that the time history of wave run-up is no longer a sinusoidal curve and the nonlinear effects are significant. The wave crest on the seaward side reaches the front location of the cylinder earlier but appears later at the location on its leeward side compared with the corresponding results when no cylinder is present. This means there is a time delay in wave propagation due to the presence of the cylinder and this phenomenon can also be observed from the free surface displacement plots at the centerline of the computational basin in the *y*-direction in Fig.17. To illustrate more clearly the deformed free surface associated with wave diffraction, the perspective and contour plots of computed free-surface displacement around the cylinder are displayed in Fig.18, from which it can be seen that the wave crest line is deformed markedly when passing through the cylinder.

Fig.19 presents the velocity distributions around the cylinder during one wave period at the centerline of the computational basin in the *y*-direction. In order to show the velocity field more clearly, the velocity field is plotted by mapping the individual particle velocities onto a fixed grid of 0.04 m× 0.02 m using the interpolations of the standard 3D spline SPH kernel. On the seaward side of the cylinder, the incident wave interacts with the radiated wave generated at the surface of the cylinder, which makes the horizontal velocities of fluid particles decrease and the vertical velocities increase. These fluid particles rise up to form an upward jet and then fall back under the action of gravity. On the landward side, the velocities of fluid particles are smaller than that on the seaward side due to the shielding effect of the cylinder.

The hydrodynamic characteristics are changed due to the presence of the cylinder. The knowledge of the velocity distributions near the seabed is helpful to understand the scouring and deposition patterns around the cylinder. Fig.20 shows the velocity and normalized turbulence eddy viscosity distributions around the cylinder at the cross section of z=0.05 m. As it can be seen in Fig.20, a fully 3D flow pattern is clearly identifiable around the cylinder. The flow velocities on the seaward and landward sides of the cylinder are smaller than that in the flank due to the blocking effect of the cylinder on the seaward side and the shielding effect on the landward side. The water particles near the seabed make a periodic reciprocating motion in the flank of the cylinder. The maximum velocity of water particles is 0.165 m/s in the flank but 0.12 m/s when no structure is present. In addition, it can be seen from Fig.20 that the normalized turbulence eddy viscosity in the flank of cylinder is greater than that on either seaward or landward side. Therefore, scouring is more likely to occur in the flank of the cylinder.

7. Conclusions

The paper presents a 3D WCSPH model to simulate wave interaction with coastal structure. The OpenMP programming technology together with an existing MPI program has been utilized to parallelize the WCSPH code. Based on this parallel WCSPH code, a numerical basin is developed and a series of validation tests are carried out to evaluate its performance.

The hybrid MPI-OpenMP implementation is shown to have much better parallel efficiency than the pure MPI programming model. The computational results also demonstrate a positive correlation between the parallel efficiency and the number of OpenMP threads within a processor.

The new SPH code has enable the development of a powerful numerical wave basin which allows a much larger wave field to be simulated than ever achieved before using SPH models. The active absorbing wave maker and the sponger layer introduced in the numerical wave basin are shown to perform well in ensuring that the waves in the numerical basin are free from the artificial re-reflected waves either from the wave maker or the downstream boundary.

In all validation tests performed excellent agreement between the numerical results and the analytical solutions or experimental data are achieved, which demonstrates that the present SPH model can be used as an effective tool to study the interactions of regular waves with impermeable marine/coastal structures. Work is currently under way to extend the model to deal with irregular waves and the permeable structures.

Acknowledgments

The improved SPH code used in this paper is developed from the open source SPHysics code developed by Prof. Robert A. Dalrymple at Johns Hopkins University, Prof. Moncho Gómez-Gesteira and Dr Alejandro Crespo at Univ.de Vigo, Dr Ben Rogers at Univ. of Manchester and others. The authors are grateful to the anonymous reviewers for their valuable comments and suggestions. This work was funded by the National Natural Science Foundation of China under Grant No. 51179030 and the Innovative Research Group National Natural Science Foundation of China under Grant No. 51221961.

References

[1] del Jesus M, Lara JL, Losada IJ. Three-dimensional interaction of waves and porous coastal structures: part I: numerical model formulation. Coastal Engineering. 2012;64:57-72.
[2] Ning D, Teng B. Numerical simulation of fully nonlinear irregular wave tank in three dimension. International Journal for numerical methods in fluids. 2007;53:1847-62.
[3] Guyenne P, Grilli S. Numerical study of three-dimensional overturning waves in shallow water. Journal of Fluid Mechanics. 2006;547:361-88.
[4] Park J, Uno Y, Sato T, Miyata H, Chun H. Numerical reproduction of fully nonlinear multi-directional waves by a viscous 3D numerical wave tank. Ocean Engineering. 2004;31:1549-65.
[5] Ha T, Lin P, Cho Y-S. Generation of 3D regular and irregular waves using Navier–Stokes equations model with an internal wave maker. Coastal Engineering. 2013;76:55-67.
[6] Higuera P, Lara JL, Losada IJ. Realistic wave generation and active wave absorption for Navier–Stokes models: Application to OpenFOAM®. Coastal Engineering. 2013;71:102-18.
[7] Higuera P, Lara JL, Losada IJ. Simulating coastal engineering processes with OpenFOAM®. Coastal Engineering. 2013;71:119-34.
[8] Monaghan JJ. Simulating free surface flows with SPH. Journal of Computational Physics. 1994;110:399-406.
[9] Gómez-Gesteira M, Cerqueiro D, Crespo C, Dalrymple R. Green water overtopping analyzed with a SPH model. Ocean Engineering. 2005;32:223-38.

[10] Shao S, Ji C, Graham DI, Reeve DE, James PW, Chadwick AJ. Simulation of wave overtopping by an incompressible SPH model. Coastal Engineering. 2006;53:723-35.
[11] Gao R, Ren B, Wang G, Wang Y. Numerical modelling of regular wave slamming on subface of open-piled structures with the corrected SPH method. Applied Ocean Research. 2012;34:173-86.
[12] Gui Q, Shao S, Dong P. Wave impact simulations by an improved ISPH model. Journal of waterway, port, coastal, and ocean engineering. 2013;140.
[13] Rogers BD, Dalrymple RA, Stansby PK. Simulation of caisson breakwater movement using 2-D SPH. Journal of Hydraulic Research. 2010;48:135-41.
[14] Didier E, Neves D, Martins R, Neves M. Wave interaction with a vertical wall: SPH numerical and experimental modeling. Ocean Engineering. 2014;88:330-41.
[15] Omidvar P, Stansby PK, Rogers BD. SPH for 3D floating bodies using variable mass particle distribution. International Journal for numerical methods in fluids. 2013;72:427-52.
[16] Bouscasse B, Colagrossi A, Marrone S, Antuono M. Nonlinear water wave interaction with floating bodies in SPH. Journal of fluids and structures. 2013;42:112-29.
[17] Ren B, He M, Dong P, Wen H. Nonlinear simulations of wave-induced motions of a freely floating body using WCSPH method. Applied Ocean Research. 2015;50:1-12.
[18] Liu X, Shao S, Lin P, Tan SK. 2D Numerical ISPH Wave Tank for Complex Fluid–Structure Coupling Problems. International Journal of Offshore and Polar Engineering. 2016;26:26-32.
[19] Akbari H, Namin MM. Moving particle method for modeling wave interaction with porous structures. Coastal Engineering. 2013;74:59-73.
[20] Ren B, Wen H, Dong P, Wang Y. Numerical simulation of wave interaction with porous structures using an improved smoothed particle hydrodynamic method. Coastal Engineering. 2014;88:88-100.
[21] Shao S. Incompressible SPH flow model for wave interactions with porous media. Coastal Engineering. 2010;57:304-16.
[22] Ren B, Wen H, Dong P, Wang Y. Improved SPH simulation of wave motions and turbulent flows through porous media. Coastal Engineering. 2016;107:14-27.
[23] Altomare C, Crespo A, Rogers B, Domínguez J, Gironella X, Gómez-Gesteira M. Numerical modelling of armour block sea breakwater with smoothed particle hydrodynamics. Computers & Structures. 2014;130:34-45.
[24] Ren B, Jin Z, Gao R, Wang Y-x, Xu Z-l. SPH-DEM Modeling of the Hydraulic Stability of 2D Blocks on a Slope. Journal of Waterway, Port, Coastal, and Ocean Engineering. 2013;140:04014022.
[25] Cummins S, Silvester T, Cleary PW. Three‐dimensional wave impact on a rigid structure using smoothed particle hydrodynamics. International Journal for numerical methods in fluids. 2012;68:1471-96.
[26] Gómez-Gesteira M, Dalrymple RA. Using a three-dimensional smoothed particle hydrodynamics method for wave impact on a tall structure. Journal of waterway, port, coastal, and ocean engineering. 2004;130:63-9.
[27] Jian W, Liang D, Shao S, Chen R, Yang K. Smoothed Particle Hydrodynamics Simulations of Dam-Break Flows Around Movable Structures. International Journal of Offshore and Polar Engineering. 2016;26:33-40.
[28] Xie J, Nistor I, Murty T. A corrected 3-D SPH method for breaking tsunami wave modelling. Natural hazards. 2012;60:81-100.
[29] Marrone S, Bouscasse B, Colagrossi A, Antuono M. Study of ship wave breaking patterns using 3D parallel SPH simulations. Computers & Fluids. 2012;69:54-66.
[30] Rogers B, Dalrymple R, Gómez-Gesteira M, Crespo A. User guide for the parallelSPHysics code using MPI v2. 0. 2011.
[31] Crespo A, Dominguez JM, Barreiro A, Gómez-Gesteira M, Rogers BD. GPUs, a new tool of acceleration in CFD: efficiency and reliability on smoothed particle hydrodynamics methods. PLoS One. 2011;6:e20685.
[32] Crespo AJ, Domínguez JM, Rogers BD, Gómez-Gesteira M, Longshaw S, Canelas R, et al. DualSPHysics: Open-source parallel CFD solver based on Smoothed Particle Hydrodynamics (SPH). Computer Physics Communications. 2015;187:204-16.
[33] Chen J, Beraun J, Carney T. A corrective smoothed particle method for boundary value problems in heat conduction. International Journal for Numerical Methods in Engineering. 1999;46:231-52.
[34] Gingold RA, Monaghan JJ. Smoothed particle hydrodynamics-theory and application to non-spherical stars. Monthly Notices of the Royal Astronomical Society. 1977;181:375-89.
[35] Wendland H. Piecewise polynomial, positive definite and compactly supported radial functions of minimal degree. Advances in computational Mathematics. 1995;4:389-96.
[36] Gotoh H, Shibahara T, Sakai T. Sub-particle-scale turbulence model for the MPS method-Lagrangian flow model for hydraulic engineering. Advanced Methods for Computational Fluid

Dynamics. 2001;9:339-47.
[37] Bradbrook K, Lane S, Richards K, Biron P, Roy A. Large eddy simulation of periodic flow characteristics at river channel confluences. Journal of Hydraulic Research. 2000;38:207-15.
[38] Gotoh H, Shao S, Memita T. SPH-LES model for numerical investigation of wave interaction with partially immersed breakwater. Coastal Engineering Journal. 2004;46:39-63.
[39] Colagrossi A, Landrini M. Numerical simulation of interfacial flows by smoothed particle hydrodynamics. Journal of Computational Physics. 2003;191:448-75.
[40] Monaghan J, Kos A. Solitary waves on a Cretan beach. Journal of waterway, port, coastal, and ocean engineering. 1999.
[41] Crespo A, Gómez-Gesteira M, Dalrymple RA. Boundary conditions generated by dynamic particles in SPH methods. CMC-TECH SCIENCE PRESS-. 2007;5:173.
[42] Oger G, Doring M, Alessandrini B, Ferrant P. Two-dimensional SPH simulations of wedge water entries. Journal of computational physics. 2006;213:803-22.
[43] Yong-xue W. Numerical wave channel with absorbing wave-maker [J]. Journal of Hydrodynamics. 1994;2:011.
[44] Gomez-Gesteira M, Rogers BD, Crespo AJ, Dalrymple R, Narayanaswamy M, Dominguez JM. SPHysics–development of a free-surface fluid solver–Part 1: Theory and formulations. Computers & Geosciences. 2012;48:289-99.
[45] Benzi J, Damodaran M. Parallel three dimensional direct simulation monte carlo for simulating micro flows. Parallel Computational Fluid Dynamics 2007: Springer; 2009. p. 91-8.
[46] Lara JL, del Jesus M, Losada IJ. Three-dimensional interaction of waves and porous coastal structures Part II: Experimental validation. Coastal Engineering. 2012;64:26-46.
[47] MacCamy RC, Fuchs RA. Wave forces on piles: A diffraction theory. DTIC Document; 1954.

Figure captions

Fig.1 Schematic plot of numerical wave basin

Fig.2 Set of periodic lateral boundary

Fig.3 Hierarchical structure of hybrid MPI-OpenMP programming.

Fig.4 Comparison of parallel efficiency for different problem sizes, each MPI processor is made up of 20 OpenMP threads, n and N are the number of particles and cores

Fig.5 Comparisons of parallel efficiency for the varying combination between MPI processors and OpenMP threads using 80 cores, n is the number of particles

Fig.6 Schematic view of the 3D SPH numerical wave basin for regular wave (T=1.2 s, H=0.12 m, d=0.5 m)

Fig.7 Comparisons of the computed and analytical water surface profiles at different measuring points (T=1.2 s, H=0.12 m, d=0.5 m)

Fig.8 Comparisons of the computed and analytical wave pressure profiles at different measuring points (T=1.2 s, H=0.12 m, x=6 m)

Fig.9 Comparisons of computed and theoretical wave velocity profiles at different measuring stations (T=1.2s, H=0.12m, x=6m)

Fig.10 Schematic plot of the numerical setup for wave interaction with vertical breakwater (Unit: m)

Fig.11 The distances between the measuring points for wave surface and the vertical breakwater (Unit: m)

Fig.12 Comparison of free surface elevation between the experimental data and SPH numerical results at different wave gauges (H=6 cm, T=2 s, d=25 cm)

Fig.13 Schematic plot of the numerical setup for wave interaction with vertical cylinder

Fig.14 Comparisons of the total forces and overturning moments acting on the vertical cylinder for case A (H=6 cm, T=1.2 s, d=0.5 m)

Fig.15 Comparisons of the total forces and overturning moments acting on the vertical cylinder for

case B (H=10 cm, T=1.2 s, d=0.5 m)

Fig.16 Wave run-up on the cylinder at the centerline of the computational basin in the y-direction: (a) upstream side and (b) downstream side

Fig.17 The free surface displacement and pressure distributions around the cylinder at the centerline of the computational basin in the y-direction (H=10 cm, T=1.2 s, d=0.5 m)

Fig.18 Perspective and contour plots of computed free-surface elevation (H=10 cm, T=1.2 s, d=0.5 m)

Fig.19 Velocity distributions around the cylinder during one wave period at the centerline of the computational basin in the y-direction (H=10 cm, T=1.2 s, d=0.5 m)

Fig.20 Velocity and normalized turbulence eddy viscosity distributions around the cylinder during one wave period at the cross section of z=0.05 m (H=10 cm, T=1.2 s, d=0.5 m)

Table caption

Table.1 The coordinates for measuring points of wave surface and pressure.